

**ADAPTIVELY REFINED EULER AND NAVIER-STOKES SOLUTIONS WITH A
CARTESIAN-CELL BASED SCHEME**

William J. Coirier
NASA Lewis Research Center
Cleveland Ohio

Kenneth G. Powell
The University of Michigan,
Ann Arbor Michigan

SUMMARY

A Cartesian-cell based scheme with adaptive mesh refinement for solving the Euler and Navier-Stokes equations in two dimensions has been developed and tested. Grids about geometrically complicated bodies were generated automatically, by recursive subdivision of a single Cartesian cell encompassing the entire flow domain. Where the resulting cells intersect bodies, N-sided "cut" cells were created using polygon-clipping algorithms. The grid was stored in a binary-tree data structure which provided a natural means of obtaining cell-to-cell connectivity and of carrying out solution-adaptive mesh refinement. The Euler and Navier-Stokes equations were solved on the resulting grids using an upwind, finite-volume formulation. The inviscid fluxes were found in an upwinded manner using a linear reconstruction of the cell primitives, providing the input states to an approximate Riemann solver. The viscous fluxes were formed using a Green-Gauss type of reconstruction upon a co-volume surrounding the cell interface. Data at the vertices of this co-volume were found in a linearly K-exact manner, which ensured linear K-exactness of the gradients. Adaptively-refined solutions for the inviscid flow about a four-element airfoil (test case 3) were compared to theory. Laminar, adaptively-refined solutions were compared to accepted computational, experimental and theoretical results.

INVISCID RESULTS

The solution procedure follows that shown in [4]. The Euler equations are solved upon a Cartesian-cell generated grid using a cell-centered, finite-volume, upwind formulation. The cell primitive variables are reconstructed using a linearly K-exact reconstruction that is slope limited, as in [2] and [3]. For the calculations shown here, Roe's [11] flux difference splitting is used as an approximate Riemann solver at the cell-to-cell interfaces. Solution adaptive mesh refinement is performed by subdividing cells according to the refinement criteria developed in [7]. This procedure computes two parameters, based upon the divergence and curl of the velocity field, which are then weighted by the cell size, 1. A simple statistical description of these parameters is then used to determine which cells

to refine and coarsen. That is, letting $\tau_c = |\nabla \bullet u| l^{3/2}$ and $\tau_r = |\nabla \times u| l^{3/2}$ represent parameters that locally describe the compressive and rotational nature of the flow field, cells are refined or coarsened if the variance of these parameters about zero is beyond some specified threshold. For the results shown here, cells are refined if

$$(\tau_c > \sigma_c \quad \text{or} \quad \tau_r > \sigma_r) \quad \text{and} \quad l > l_{min} \quad (1)$$

and cells are coarsened if

$$\tau_c < \frac{\sigma_c}{10} \quad \text{and} \quad \tau_r < \frac{\sigma_r}{10} \quad (2)$$

Experience ([4] and [6]) has dictated the one-tenth scaling of the variance for coarsening, and in practice, the minimum allowable refineable cell size in (1) is typically taken to be 0.001 chords. In all of the adaptively-refined computations shown here, the refinement criteria is set exactly as above. For simplicity, a three-stage, multi-stage scheme is used to advance the equations in pseudo-time, with stage coefficients $\lambda = (0.18, 0.5, 1.0)$. A spatially varying time-step is formed using blended hyperbolic and parabolic stability constraints.

Test Case 3: Suddhoo-Hall Four-Element Airfoil

This test case geometry corresponds to that shown in [13] where successive Karman-Trefftz transformations were applied to a series of circles in the complex plane, resulting in a high-lift-like set of four-element airfoil shapes. The geometry has been approximated using the workshop supplied cubic-splines, and adaptively refined solutions made using the Cartesian, cell-based approach. The free stream Mach number is $M_\infty = 0.2$ and the angle of attack is $\alpha = 0^\circ$. Three levels of adaptive-mesh refinement were made beyond the base grid level. The computed surface pressure coefficients for all the refinement levels are shown along with the geometry in Figure 1. The variation of the computed lift and drag coefficients through the adaptive mesh refinement is shown in Figure 2. Computations using the Cartesian approach on a selection of the inviscid test cases are shown in a companion paper [10].

VISCOUS SOLUTIONS

In [5] and [4] adaptively refined solutions of the Navier-Stokes equations using a Cartesian, cell-based approach are shown for a selection of low and moderate Reynolds number flows. The viscous fluxes are found upon each cell interface using a Green-Gauss type of reconstruction performed about a co-volume located about the interface [4]. The data at the vertices of this co-volume are found in a linearity preserving manner ([4] and [9]), which guarantees the linear K-exactness of the reconstructed gradients. To demonstrate the approach, a selection of the results computed in [4] are shown here.

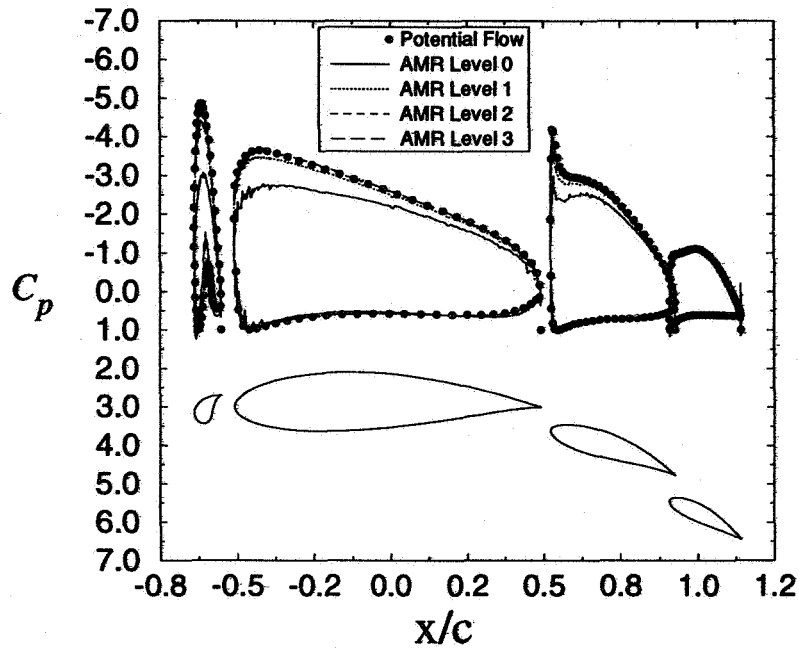


Figure 1 Computed surface pressure coefficient data through adaptive mesh refinement.

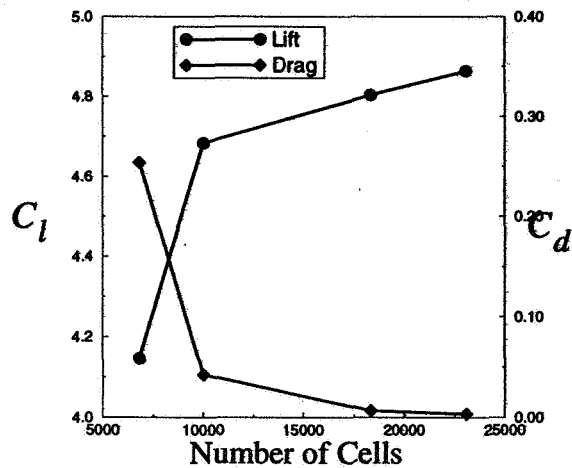


Figure 2 Computed lift and drag coefficients.

Laminar, Driven Cavity Flow: $Re=100$ and $Re=400$

The laminar flow inside a square driven cavity is computed and compared to the computed results of Ghia[8]. In [8], an incompressible formulation of the Navier-Stokes equations was solved using an implicit multi-grid method, where tabulated u- and v-velocity data were supplied

along the lines through the geometric center of the cavity. To compare with these incompressible results, the Mach number used here is taken to be $M_{lid} = 0.1$. For the $Re=100$ case, a uniform base grid of 1024 cells (32 by 32) is generated, and three levels of adaptive mesh refinement beyond the base grid are obtained. Adaptive mesh refinement improves the solution slightly, but the initial solution is quite good. Figure 3 shows the computed u - and v -velocity profiles along vertical and horizontal lines through the geometric center of the cavity for the $Re=100$ case. For the $Re=400$ case, the initial solution is poor, but the adaptive-mesh refinement improves the solution quality with each successive level of refinement, until an acceptably good solution is obtained at the final refinement level. Figure 4 shows the computed u - and v -velocity profiles through mesh refinement.

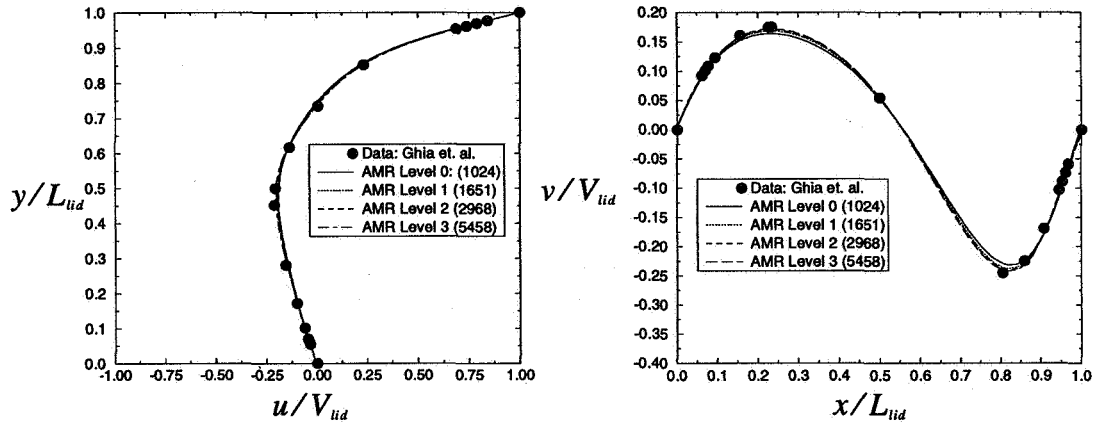


Figure 3 Computed u - and v -velocities through adaptive-mesh refinement for the $Re=100$ driven cavity problem.

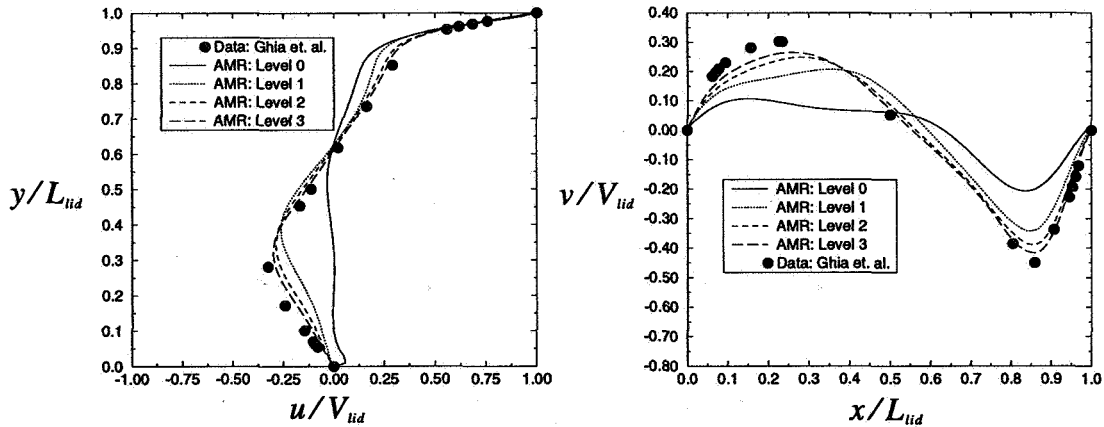


Figure 4 Computed u - and v -velocities through adaptive-mesh refinement for the $Re=400$ driven cavity problem.

Laminar Flow Over a Backward Facing Step

The laminar flow over a backwards facing step at two Reynolds numbers is used to validate the solver in [4]. The computed results are compared to the experimental data of [1] at the Reynolds numbers of $Re=100$ and $Re=389$. A parabolic velocity profile is specified at the inflow, and the exit pressure is specified. This ensures that the proper pressure gradient is imposed on the flow. A coarse base grid is generated, and adaptive mesh refinement is made for three subsequent levels of refinement for both Reynolds numbers. Figure 5 shows the effect of adaptive mesh refinement at a location corresponding to 2.55 step heights downstream of the step. Comparisons are made at other locations of the flow in [4]. The results compare well, and are not shown here. The agreement with the experimental data is good, and the adaptive mesh refinement improves the solution quality with each refinement.

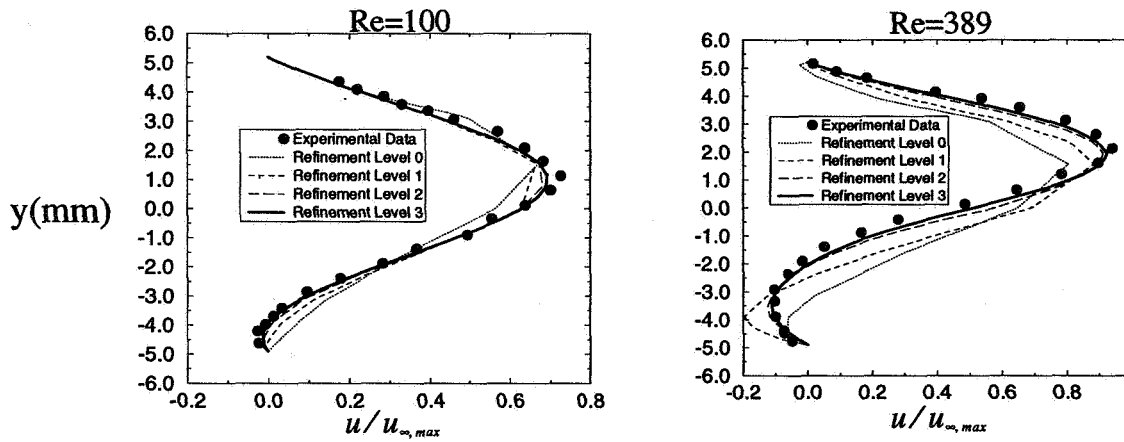


Figure 5 Computed u -velocities at 2.55 step heights beyond step for Reynolds numbers $Re=100$ and $Re=389$.

Laminar, Developing Boundary-Layer Flow

The laminar flow over a flat plate which is aligned with the free stream is computed with the Cartesian solver, and compared to theory. Uniform flow is imposed ahead of the plate leading edge, and the boundary-layer develops to a location where the Reynolds number based on distance from the leading edge is $Re_x = 10,000$. The effect of the introduction of cut cells, with their inherent non-smoothness, is illustrated by computing this flow on two series of grids. The first grid sequence is created by orienting the base axes of the Cartesian system coincident with the plate surface, yielding a base grid with no cell cutting, which is then adaptively-refined. The second grid sequence is created by rotating the plate surface 30° with respect to the x -axis, introducing many cut cells along the plate boundary, which also is adaptively refined.

For the axes-aligned cases, when sufficient resolution is supplied, the mean flow profiles com-

pare very well with theory, although the skin friction exhibits small scale oscillations whenever a refinement boundary is located near the wall. Figure 6 shows the computed u- and v-velocity profiles at a location corresponding to $Re_x = 8000$.

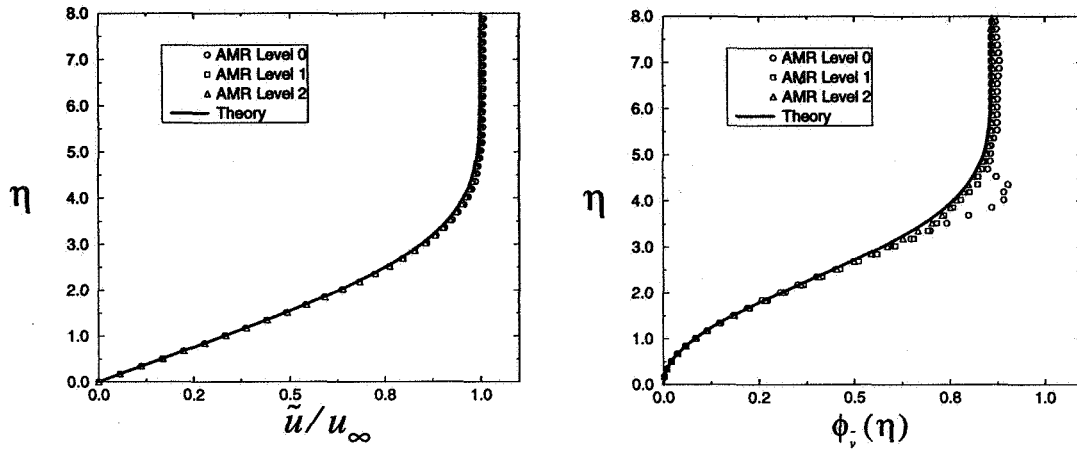


Figure 6 Predicted u- and v-velocity profiles for axes-aligned flat plate.

The grid non-smoothness induced on the non-axes aligned grid caused convergence problems, which was alleviated by using a local modification to the viscous gradient reconstruction procedure in cut cells and their neighbors. The computed u- and v-velocity profiles, shown in Figure 7, compared moderately well to theory, but the skin friction exhibit large scale oscillations.

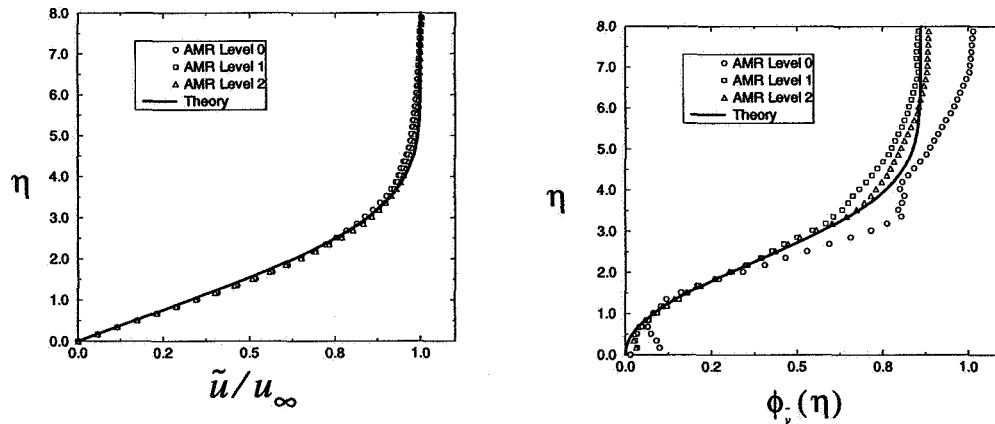


Figure 7 Predicted u- and v-velocity profiles for non-axes-aligned flat plate

The oscillations induced by the extreme grid non-smoothness caused by the cut cells is indicative of the sensitivity of current viscous flux functions to grid smoothness. This sensitivity is highlighted by the grids generated using the Cartesian approach, since extremely non-smooth grids are created near walls, where the shear is typically high. The result of this is typically oscillations in the skin friction and heat transfer rates, and due to the non-positivity of the viscous operators, the convergence can be adversely effected. Regardless of these negative findings, the approach can still prove useful in per-

forming automated grid generation and adaptive mesh refinement upon more geometrically and dynamically complicated flows, as is shown in the next example.

Simulated Branched Duct

To demonstrate the approach for complex geometries, the flow in a stylized duct is computed. This duct geometry corresponds to an experiment conducted at NASA LeRC designed to simulate, in a simplified manner, the flow in the cooling passages of a turbine blade [12]. The calculations shown here in no way try to simulate the experiment: The experimental conditions correspond to a turbulent flow, while the calculations shown here are laminar. A fully developed profile is introduced at the inflow, and the flow is diverted into the primary passage by the blockage introduced by the pin fins in the secondary passage. The Reynolds number based on maximum inflow velocity and pin fin diameter is $Re=25$. Only one level of adaptive-mesh refinement beyond the base grid level is obtained, due to positivity problems in the rear stagnation region of one of the pin fins. The final adapted grid and contours of total velocity are shown in Figure 8 and Figure 9.

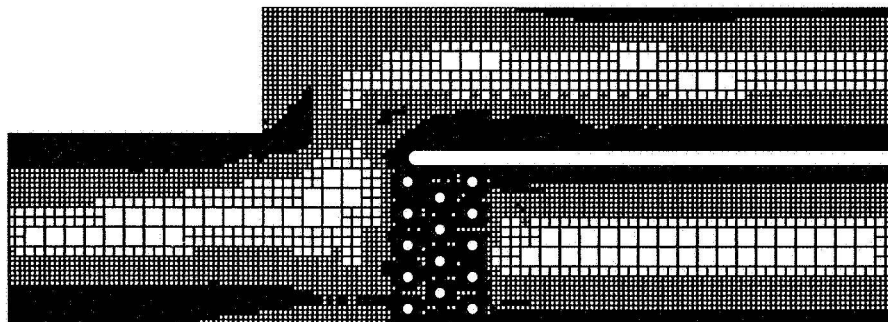


Figure 8 Adapted grid, branched-duct.

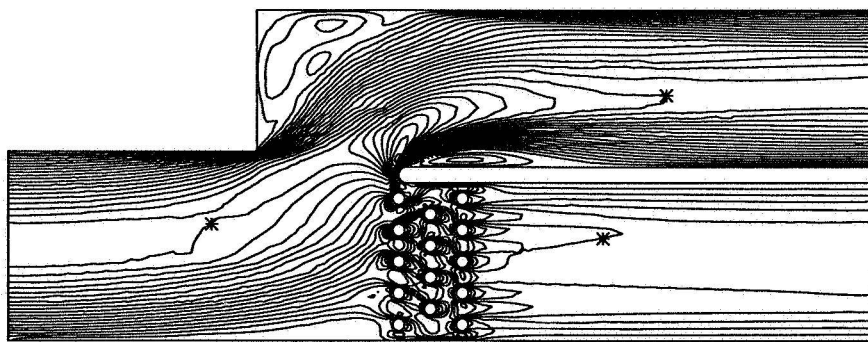


Figure 9 Computed total velocity contours.

The basic flow features predicted here correspond to those in the experiment, although some important features are grossly under-resolved, such as the individual pin-fin wakes. The primary passage separation and reattachment along the splitter plate and the separation anchored at the back step

portion are both properly predicted, as well as the upstream influence of the pin blockage upon the lower wall flow. Although many levels of refinement were not achieved, the larger scale flow features were adequately predicted and there resolution was improved by the mesh refinement procedure.

CONCLUSIONS

Adaptively-refined solutions of the Euler and Navier-Stokes equations using a Cartesian, cell-based approach have been made. Inviscid computations corresponding to test case 3 of the workshop compared favorably with theory. The emphasis here has been upon the extension of the Cartesian, cell-based method to computing viscous flows. Adaptively-refined solutions of the Navier-Stokes equations have been made, and the results compared well to accepted computational, experimental and theoretical data. An inherent weakness of the Cartesian approach is brought forth, that is directly tied to one of the properties that makes the approach useful: The Cartesian approach sacrifices grid smoothness for automation of the mesh generation. This grid non-smoothness is not handled well by the current generation viscous flux functions, which tend to produce non-positive and inaccurate stencils upon distorted grids. Regardless of this comparatively negative finding, the approach has proven to be useful, and can provide accurate, automatically meshed and adaptively-refined solutions of the Euler and Navier-Stokes equations.

REFERENCES

- [1] B.F. Armaly, F. Durst, J.C.F. Pereira, and B. Schonung. Experimental and Theoretical Investigation of Backward-Facing Step Flow. *Journal of Fluid Mechanics*, 127:473–496, 1983.
- [2] T. J. Barth and P. O. Frederickson. Higher Order Solution of the Euler Equations on Unstructured Grids Using Quadratic Reconstruction. AIAA Paper 90-0013, 1990.
- [3] T.J. Barth and D.C. Jespersen. The Design and Application of Upwind Schemes on Unstructured Meshes. AIAA Paper 89-0366, 1989.
- [4] W.J. Coirier. *An Adaptively-Refined, Cartesian, Cell-Based Scheme for the Euler and Navier-Stokes Equations*. PhD thesis, The University of Michigan, Department Aerospace Engineering. Also published as NASA TM 106754, 1994.
- [5] W.J. Coirier and K.G. Powell. A Cartesian, Cell-Based Approach for Adaptively-Refined Solutions of the Euler and Navier-Stokes Equations. AIAA paper AIAA-95-0566, 1995.
- [6] D. DeZeeuw and K.G. Powell. Euler Calculations of Axisymmetric Under-Expanded Jets by an Adaptive-Refinement Method. AIAA Paper 92-0321, 1992.
- [7] D.L. DeZeeuw. *A Quadtree-Based Adaptively-Refined Cartesian-Grid Algorithm for Solution of the Euler Equations*. PhD thesis, The University of Michigan, Department of Aerospace Engineering, 1993.
- [8] U. Ghia, K.N. Ghia, and C.T. Shin. High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method. *Journal of Computational Physics*, 48:387–411, 1982.
- [9] D.G. Holmes and S.D. Connell. Solution of the 2D Navier-Stokes Equations on Unstructured

Adaptive Grids. AIAA Paper 89-1932-CP, 1989.

- [10] Kenneth G. Powell. The Adaptive Cartesian-Grid Approach, Warts and All. In *Proceedings of the ICASE/LaRC Workshop on Adaptive Grid Methods*, 1995.
- [11] P. L. Roe. Approximate Riemann Solvers, Parameter Vectors and Difference Schemes. *Journal of Computational Physics*, 43, 1981.
- [12] L.M. Russell, D.R. Thurman, P.S. Simonyi, S.A. Hippensteele, and P.E. Poinsatte. Measurements and Computational Analysis of Heat Transfer and Flow in a Simulated Turbine Blade Internal Cooling Passage. AIAA Paper AIAA-93-1797, 1993.
- [13] A. Suddhoo and I.M. Hall. Test Cases for the Plane Potential Flow past Multi-Element Aerofoils. *Aeronautical Journal*, pages 403–414, 1985.